GRAVITY-CURRENT TRANSPORT IN BUILDING FIRES

by

H.R. Baum, K.W. Cassel,
K.B. McGrattan, and R. G. Rehm
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

International Conference on Fire Research and Engineering, September 10-15, 1995. Orlando, FL. Proceedings. Sponsored by National Institute of Standards and Technology (NIST) and Society of Fire Protection Engineers (SFPE). D. Peter Lund and Elizabeth A. Angell, Editors. Society of Fire Protection Engineers, Boston, MA, 1995.

Gravity-Current Transport in Building Fires

H.R. Baum, K.W. Cassel, K.B. McGrattan and R.G. Rehm
National Institute of Standards and Technology

1 Introduction

When lighter fluid is introduced at the top of an ambient fluid (or heavier fluid at the bottom of ambient fluid), the light fluid spreads over the ambient, forming a gravity current (GC). Gravity currents are well-known in the geophysical literature and have been studied both experimentally and theoretically for many years [1], [2].

Gravity currents are also of interest in the movement of gases in buildings. When considering the flow of air due to heating and ventilating systems and in the study of the spread of smoke and hot gases due to fires in buildings, gravity currents play a very important role. A GC produced by a fire can transport smoke, toxic material and hot gases, and when the building has long corridors, the current often is one of the most important mechanisms for large-scale mass and energy transport. Furthermore, the transit time for a GC in a corridor can have major impact on the egress time from the structure. Heat transfer is a critical issue and one that has not been addressed adequately to our knowledge. During the past several years, the authors have developed mathematical models and algorithms which describe the buoyant convection induced by a fire in an enclosure [3], [4]. It is now possible to compute the structure of GCs in detail and to compare features of GCs with available experimental and analytical results.

In the next section, the authors describe GCs by solving the Navier-Stokes equations in two dimensions. We make comparisons between results from salt-water, fresh-water experiments carried out by Zukoski and coworkers [5], [6] and the results obtained from our computational simulations. Because the model is restricted to two dimensions, very high resolution computations can be performed, and these computations allow us to resolve both large-scale features and small-scale friction and heat transfer for Reynolds numbers of interest.

2 Hydrodynamic Model

We consider the flow of a Boussinesq fluid in a rectangular enclosure, with density differences induced either by temperature or by concentration variations. To compare with experiments, we also consider the latter case; there is no difference in the dynamics between the two if proper account is taken of the sign (and size) difference between thermal expansion and density increase with salinity, as well as the differences in magnitude of the transport coefficients. Cool gas or saline water spread along the floor with exactly the same dynamics as heated gas spreads under a ceiling, provided that there is no heat transfer (the case we examine first for comparison). Here, we compare our computations with experiments [5] [6] in which a flow is induced by the introduction of salt water into a long rectangular tank filled with fresh water. The salt water flows into the tank via a slot opening at the bottom at one end, and an equal volume of overflow is evacuated from the same sized opening at the top of the other end. For additional details on these comparisons, see [4] and [7]. Typical computed profiles of the current are shown in Figure 1.

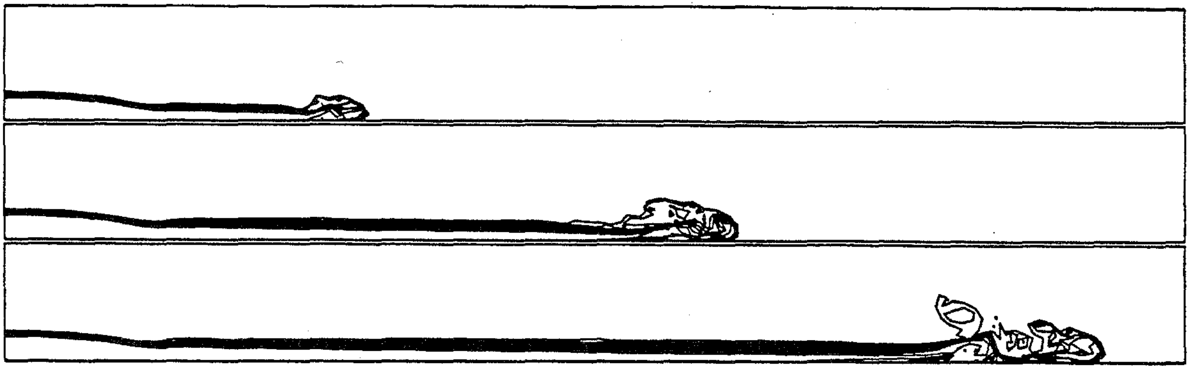

Figure 1: A gravity current develops as heavy fluid is pumped into the channel at the lower left. Fluid is evacuated at the upper right.

The equations of motion for a Boussinesq fluid are

$$\begin{aligned} \operatorname{div}\mathbf{u} &= 0 \\ \partial(T)/\partial t + \operatorname{div}(T\mathbf{u}) &= \kappa\nabla^2 T \\ \rho\,(\partial\mathbf{u}/\partial t + \mathbf{u}\cdot\nabla\mathbf{u}) + \nabla p - \rho\mathbf{g} &= \rho\nu\nabla^2\mathbf{u} \end{aligned} \tag{1}$$

where T is the temperature, $\mathbf{u}$ is the velocity vector, p is the pressure, $\mathbf{g}$ is the acceleration of gravity, ν is the kinematic viscosity, and κ is the thermal diffusivity. The latter two quantities will be assumed constant. The density and the temperature are related by an isobaric equation of state $p_0 = \rho RT$ where R is the gas constant.

We employ the same scalings as used by Zukoski in his experiments [5]. All lengths are measured relative to the height h of the GC, all velocities are relative to the characteristic velocity $U = \sqrt{\beta g h}$, and time is relative to h/U. The characteristic length and velocity scales are related to the flow rate by $Q = Uh$. The total density of the mixture of can be expressed as $\rho = \rho_0(1 + \beta\tilde{\rho})$, where $\beta = (\rho_1 - \rho_0)/\rho_0$; and ρ_0 and ρ_1 are the densities of the fresh and salt water or ambient and heated (or cooled) gas, respectively. The pressure can be written $p = p_0 + \tilde{p}$ where p_0 satisfies the hydrostatic condition $\nabla p_0 = \rho_0\mathbf{g}$.

We assume the relative density difference $\beta \ll 1$ (the Boussinesq approximation), *i.e.*, there are only small temperature differences in the heated-gas case. A prescribed flux of fluid of specified density is introduced along a segment of the bottom boundary, and, since the fluid is incompressible an equal amount of fluid must be extracted elsewhere.

Eqs. (1) are a mixed parabolic/elliptic system of partial differential equations, *i.e.*, the equations for the density and the velocity components are parabolic, whereas the equation for the pressure is elliptic. The spatial grid is taken to be uniform in each of the two directions, although the mesh length may be different in each direction. Within each mesh cell, a rectangle, vector components are evaluated at the sides and scalar quantities at the center. The staggered grid permits central differences to second-order accuracy for all linear operations. The flow variables are updated in time according to a simple second-order Runge-Kutta scheme.

The pressure equation is formed by taking the divergence of the momentum equations and enforcing the first of Eqs. (1). The linear algebraic system arising from its discretization has constant coefficients and can be solved by a fast direct method, see [8] for details. The solution to the pressure equation constitutes the bulk of the numerical computation since the density and the velocity are updated explicitly once the pressure gradients are known.

The resolution of the computation determines the maximum Reynolds number of the flow which can be calculated. The size of the Reynolds number for the 2-D flow scales as the square of the number of grid cells in the vertical direction. The algorithm will fail if the Reynolds number is too large for the resolution of the grid. This feature, plus the fact that there are no adjustable parameters in the code, lead us to have confidence in the predictive capability of the computations.

3 Results and Discussion

Although there is a distinct three-dimensional nature to GCs [1], much of the large-scale structure can be described as two dimensional. One of the objectives of this study was to examine how well a one-dimensional model based on shallow-water theory or a two-dimensional model based on direct numerical integration of the Navier-Stokes equations can describe large-scale features of GCs. In this section comparison between the two-dimensional model and experimental results are made, while examination of the one-dimensional model is made in the next section.

To compare the numerical simulations with the experiments quantitatively, we plot the front trajectories for both. The experiment is conducted in a tank 274 cm long, 15 cm high, and 15 cm wide. Salt water is pumped into the tank through a slot 10 cm wide at a rate Q =17 cm^2/s. The quantity $g\beta$ is given as 134 cm/s^2. Since $Q = Uh = \sqrt{g\beta h}h$, we determine the height of the current h to be 1.29 cm, the ratio H/h to be 11.6, and the Reynolds number Q/ν to be 1697. For the computation, we use somewhat different conditions to reduce computational resource requirements; these compromises do not degrade the trajectory comparison however. We use a 16×1 rectangular enclosure (3072×192 cells); a ratio H/h of 8 (smaller ratios of H/h begin to show an influence of the upper boundary on the GC); a computational Reynolds number somewhat less than the Reynolds number required by the experiments; and a Schmidt number of order unity. (The Schmidt number does not affect the outcome. A computation with no mass diffusion, the infinite-Schmidt-number case, can be computed somewhat differently [4]. The trajectories computed in each case are practically indistinguishable.) Figure 2 is a comparison of the trajectories for the experiment and the simulation. The reduced tank and H/h ratios and the reduced Reynolds number from the experiments do not seem to affect the trajectories.

There have been relatively few studies concerning viscous effects on GCs. Some early studies are cited in the book of Simpson [1]. Didden and Maxworthy [9], using dimensional

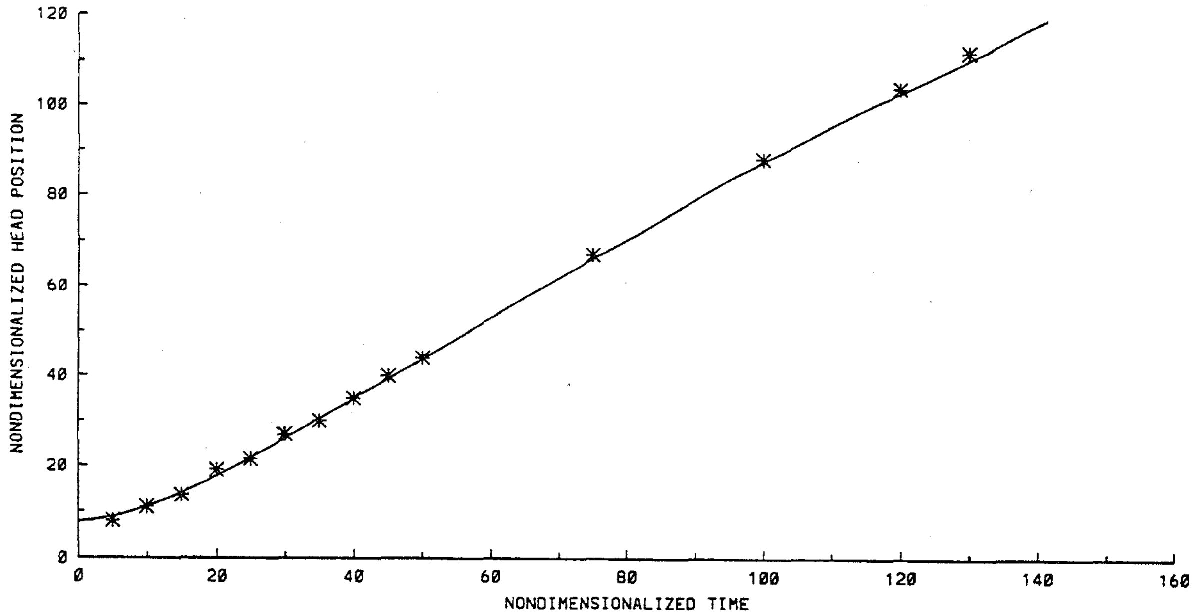

Figure 2: A comparison of a computed gravity current trajectory and Zukoski's experimental data (stars)

and order-of-magnitude arguments to determine the governing time scales, established conceptually the regimes of flow of a viscous GC. The first regime is determined by a balance between gravity and inertia (the inertia-buoyancy regime, [11]), and the later regime is determined by a balance between viscous spreading and buoyancy effects (the viscous-buoyancy regime). Experimental verification for this study was based on limited results, mostly in the axisymmetric case. Other studies [10], [11], [5] and [6], present analytical, numerical and experimental results within this conceptual framework. Unfortunately, while the division into these regimes is reasonable conceptually, it is of less value practically, as determined both by the experimental data of Zukoski and by our detailed numerical computations (see Figure 2). The difficulty is in the fact that the intermediate regime, in which all three forces (inertia, buoyancy and viscous) are important, persists over much of the time for which practical measurements and computations can be made.

Next, we consider some effects of heat transfer on a GC generated by a hot layer of temperature T_1 flowing into a corridor initially at temperature T_0. Figure 3 shows temperature contours and local ceiling heat transfer coefficients as the GC progresses down a corridor. In these composite plots, the upper curve shows the Nusselt number $Nu = q_w(x)H/k(T_1 - T_0)$ at times $t = 30$ and 50 superimposed over the corresponding temperature contours. Here q_w is the local heat flux to the ceiling and k the thermal conductivity of the gas. For this computation, the parameters are the same as for Figure 2 but with $\beta = (T_1 - T_0)/T_0$ and a Prandtl number of 0.7. The wall is assumed to remain cold, providing the maximum heat transfer. Note that the heat transfer profiles are not smooth, but exhibit large variation with axial distance, particularly at the left (the inlet) and at the head of the GC. The heat transfer is very large at the inlet because the gas is the hottest there. In the head region, the mixing is greatest, causing large and intermittent heat transfer; the vortices shed by the

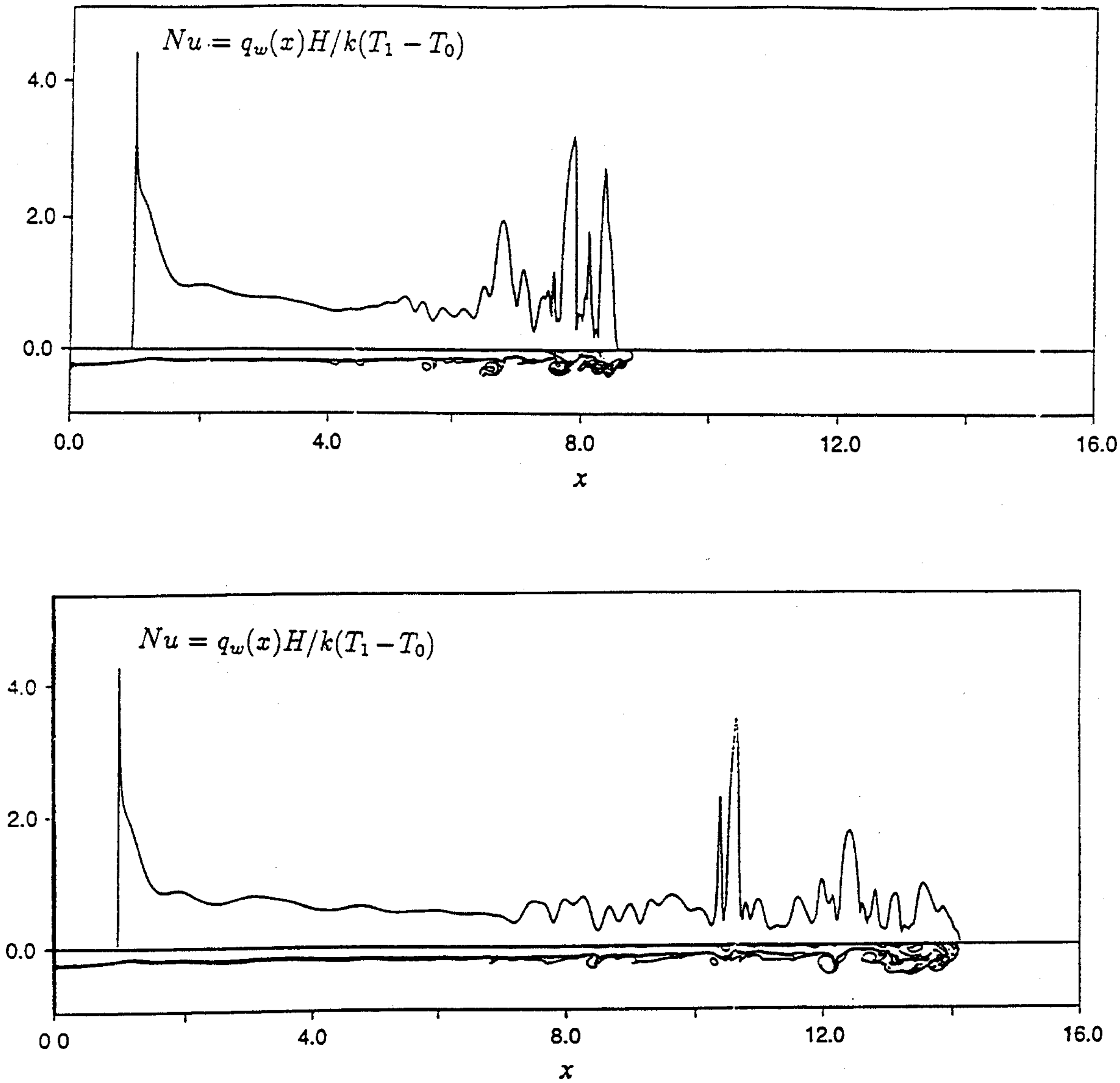

Figure 3: Plots of local Nusselt number $Nu = q_w(x)H/k(T_1 - T_0)$ and corresponding temperature contours at dimensionless times $t = 30$ (top) and $t = 50$. Here, x is the axial distance down the corridor from the left wall. Inflow of hot gases occurs along the top wall for a distance of one dimensionless unit in x.

head produce large transient heat transfer in agreement with the analysis reported by two of the authors in another study [12]. Finally, a comparison of the location of the head at each time with what was determined in the absence of heat transfer (Figure 2), shows that the heat transfer substantially slows the progression of the GC.

References

[1] Simpson, J.E., *Gravity Currents in the Environment and the Laboratory*, Halsted Press: a division of John Wiley & Sons, New York, 1987.

[2] Turner, J.S., *Buoyancy Effects in Fluids*, Cambridge University Press, Cambridge, 1973.

[3] Rehm, R.G. and H.R. Baum, "The Equations of Motion for Thermally Driven, Buoyant Flows", *Journal of Research of the NBS*, Vol. 83, pp 297-308, May-June 1978.

[4] McGrattan, K.B., R.G. Rehm and H.R. Baum, "Fire-Driven Flows in Enclosures," *Journal of Computational Physics*, Vol. 110, No. 2, pp 285-291, 1994.

[5] Zukoski, E.E., "Scaling Rules for Smoke Movement in Corridors", June 1990, and data on experiments. Private communication.

[6] Chan, W.R., E.E. Zukoski and T. Kubota, "Experimental and Numerical Studies of Two-Dimensional Gravity Currents in a Horizontal Channel", Guggenheim Jet Propulsion Center Report, Cal. Tech., Pasadena, California.

[7] Baum, H.R., O.A. Ezekoye, K.B. McGrattan and R.G. Rehm, "Mathematical Modeling and Computer Simulation of Fire Phenomena," *Theoretical and Computational Fluid Dynamics*, Vol. 6, pp 125-39, Springer-Verlag, 1994.

[8] Baum, H.R. and R.G. Rehm, "Calculations of Three-Dimensional Buoyant Plumes in Enclosures", *Combustion Science and Technology*, Vol. 40, pp 55-77, Gordon and Breach Science Publishers, 1984.

[9] Didden, N. and Maxworthy, T., "The Viscous Spread of Plane and Axisymmetric Gravity Currents," J. Fluid Mech., bf 121, pp. 27-42, 1982.

[10] Huppert, H.E., "The Propagation of Two-Dimensional and Axisymmetric Viscous Gravity Currents Over a Rigid Horizontal Surface," J. Fluid Mech., bf 121, pp. 43-58, 1982.

[11] Nakos, J.T. and Wildin, M.W., "Prediction of Velocity and Temperature Profiles in Thermally Driven Gravity Currents Applied to Stratified Thermal Storage Tanks," AIAA, ASME, SIAM, APS First National Fluid Dynamics Congress, Collection of Technical Papers, Cincinnati, Ohio, July 25-28, 1988, pp. 2027-2035.

[12] Cassel, K.W. and R.G. Rehm, "The Unsteady Mixed-Convection Boundary Layer Induced by a Vortex," Paper submitted to The Eighth International Symposium on Transport Phenomena in Combustion, San Francisco, July 16-20, 1995.